SIMULATION OF LEAK FLOW RATE THROUGH CIRCUMFERENTIAL CRACKS FROM HIGH PRESSURE PIPELINES

Abstract.

In this work mass flux through the outlet of a crack has been examined for circumferential cracks of circular and rectangular shapes. Computational hydrodynamic simulation of narrow leaks in a pipe having a 100 mm diameter has been computed using a 3-dimensional k-epsilon flow model through ANSYS FLUENT 2023 software. Variations of fluid flow in mass flux and velocity through three different sizes of circular and rectangular cracks are studied separately at different inlet pressures. The results articulate that the due to the presence of substantial changes in the flow properties can happen with varying mainstream pressures. The results obtained possibly will contribute towards leak flow analysis for various industrial and commercial purposes.

Keywords: Mass flux, Circumferential cracks, Circular cracks, Rectangular cracks, Narrow leaks, Pipe flow

Authors

Anwar Hussain
Mechanical Engineering Department,
JIS College of Engineering, Kalyani,
West Bengal 741235, India

Sandip Ghosh
Mechanical Engineering Department,
JIS College of Engineering, Kalyani,
West Bengal 741235, India

Sanjiban Biswas
Mechanical Engineering Department,
JIS College of Engineering, Kalyani,
West Bengal 741235, India

Bidyut Karmakar
Mechanical Engineering Department,
JIS College of Engineering, Kalyani,
West Bengal 741235, India

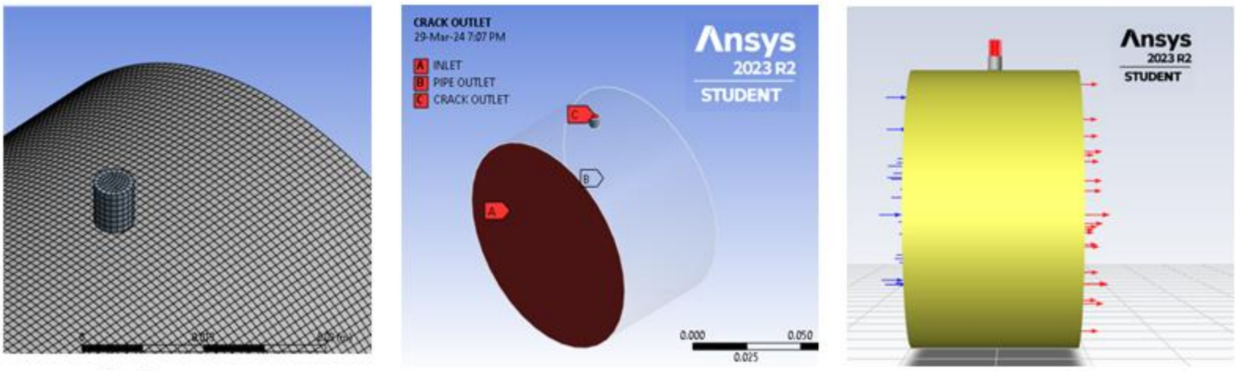
Sreeparna Das
Mechanical Engineering Department,
JIS College of Engineering, Kalyani,
West Bengal 741235, India

Soumik Ghosh
Mechanical Engineering Department,
JIS College of Engineering, Kalyani,
West Bengal 741235, India

Rahul Karmakar
Mechanical Engineering Department,
JIS College of Engineering, Kalyani,
West Bengal 741235, India

I. INTRODUCTION

High pressure piping systems is a part and parcel of power and process industries throughout the world. Transport of different fluid through pipelines across long distances has gained momentum since last few decades because of cost effectiveness, safety and huge volume transport in chemical industry, power industry, water supply plants etc. Leakage of the transported fluid is an inevitable phenomenon which becomes a great economic and environmental concern. Pipeline leakages may occur due to sudden change in pressure, cracks, corrosive action, defect in pipes, bad workmanship, any destructive causes as well as lack of maintenance. Several literature and field studies [1] have reported that pressure reduction is advantageous in controlling the leakage and shown significant increase in leakage with an increase in pressure. An experimental investigation have reported that for a given pressure differential across the orifice, the mainstream velocity shows significant impact with increasing outflow velocity as the primary flow velocity increases. Around 23% increases in leakage was monitored for a change in pressure from 1 to 7 bars [2]. Another study using leakage openings shaped as orifice revealed that by increasing the diameter of the leak orifice, the volume of the fluid leaving the orifice was greater and this affects the hydrodynamics in the neighborhood of the leak inside the pipe [3]. It was evident from the claims that monitoring the pressure and velocity is crucial to identify the leak and any chance of catastrophic breakage can be ruled out using modern leak detection techniques. Computational methods have revolutionalized the thermal hydraulic analysis of high pressure high temperature pipelines simulating the industry environments which has emerged as highly cost effective technique in comparison to their experimental counterparts. An analysis carryout out using COMSOL multiphysics platform concluded that the pressure and velocity distributions near the crack were affected by the crack geometry and the trends were found to be similar for circular and square cracks [4]. More computational fluid dynamics simulation using FLUENT code has described the change in flow behavior in a pipeline with a circumferential crack on stainless steel pipes [5] and demonstrated the effect of leakage on the pressure and velocity distribution across the leak flow. A detailed analytical investigation has reported the modeling and simulation of the loss of coolant accident in nuclear power plants through circumferential pipe cracks [6]. In the analysis, it was determined that examining flow dynamics through cracks across a range of stagnation conditions is beneficial to predict leak flow patterns in two phase high speed critical flow condition. This study also provided insights into enhancing safety analyses for high-energy pipelines utilized in nuclear power facilities. More investigations done on small leaks in water pipelines have been conducted using computational fluid dynamics simulations [7] to analyze the effects of variation in pressure at lower range of pressures. The work focused on investigating a steady-state 3D turbulent model, examining various crack sizes in the pipelines. The findings indicate that leaks significantly impact the pressure gradient. A probabilistic analysis using leak before break methodology, employing the failure assessment diagram has been reported [8], which predicts the failure probabilities for various crack types through Monte Carlo simulation. However, the findings underscored the significant influence of crack morphology parameters on leak rates. A significant number of researches have focused on the leakage rate calculations based on experimental and analytical methods for single and two phase leak flow conditions [9-12]. The present study investigates mass flux and velocities through cracks of various shapes and sizes under varying pressure gradients. Comparable findings have been documented in numerous other papers. This work endeavors to improve the detection of pipe crack leakage within high-pressure environments, thereby facilitating advancements in the field as a whole.



Meshed geometry **Pipe inlet, pipe outlet and crack outlet** **Computational domain**

Figure 1. Physical model of the leakage geometry

II. PHYSICAL PROBLEM

1. Modelling of the Computational Domain

In the present work, a study has been conducted using a pipe of 100 mm length and 100 mm diameter. Artificial cracks have been generated. Three dimensional models have been developed in the design module of ANSYS workbench platform. Normal water and its standard properties are considered as the fluid under leak flow condition and wall conditions are designated with the properties of carbon steel. The major flow diameter was taken as 100 mm with a pressure gradient of 1 bar between the inlet and outlet for all crack dimensions. Rectangular cracks of sizes 1mm x 2mm, 1mm x 5mm, and 1mm x 8mm and circular cracks of diameters 1mm, 3mm, and 5mm were created. Inlet pressures were taken as 70, 80, 90 and 100 bar. A study has also been conducted using a pipe of 500 mm length and 100 mm diameter. Cracks have been generated in an open source CFD simulation platform in the similar way. Same pressure gradient of 1 bar between the inlet and outlet of the main flow for all the cases that is for all the crack dimensions. Rectangular cracks of sizes 0.5mm x 1mm, 0.5 mm x 2 mm, and 0.5 mm x 3 mm and circular cracks of diameters 1.5 mm, 2 mm, and 2.5 mm were created. Inlet pressures were taken as 110, 120 and 130 bar. Mesh generation and refinement was conducted by adjusting parameters such as element size and number of divisions while validating computational setup. Discretization is achieved through the appropriate model selections as referred by allied researchers. Mass flow rate was determined by solving continuity and momentum equation using FLUENT code and appropriate turbulence model. Standard K-epsilon model was considered for the present analysis with different pressure inlet values and a constant pressure gradient for different crack dimensions. The data has been used to compute mass flux (Kg/m^{2}s) and maximum velocities at crack outlets for analysis.

2. Boundary Conditions

A pressure-based standard k-epsilon turbulence model has been employed in the simulation, utilizing double precision. The inlet pressure has been adjusted while maintaining a constant outlet pressure through a crack. The outlet pipe pressure has been set to uphold a consistent pressure gradient for the primary pipe flow. The governing equations for describing the fluid flow within the pipe and leakage entail the conservation of mass and momentum. The fluid properties are considered Newtonian and incompressible, with constant physical and chemical attributes. Moreover, the flow is assumed to be three-dimensional and isothermal. The specific equations characterizing the flow are delineated below.

The equation for momentum conservation in the i direction can be expressed as-

$$\frac{\delta}{\delta t}(\rho \bar{U_i}) + \frac{\delta}{\delta x_j}(\rho U_i U_j) = -\frac{\delta p}{\delta x_i} + \frac{\delta}{\delta x_j}\left(\mu \frac{\delta U_i}{\delta x_j}\right) \quad (1)$$

The continuity equation exhibits the following form at the leak location:

$$Q_{up} - Q_{down} - Q_{leak} = 0 \quad (2)$$

Where, Q_{up} and Q_{down} is the discharge upstream and downstream of leak location respectively. Q_{leak} is the flow rate of the leak. Pressure drop across the test section is determined by the Darcy-Weisbach equation

$$\Delta P = L \times f_D \times (\rho / 2) \times (v^2 / D) \quad (3)$$

where, L = the length of the pipe [m], D = pipe diameter in m, Δp = pressure loss [Pa], ρ = fluid density [kg/m^3], f_D= Darcy friction factor [unitless], v = mean fluid velocity [m/s], friction factor f_D = 64 / R_e, where R_e is Reynolds number.

III. RESULT AND DISCUSSION

For circular cracks the predicted mass flux and maximum leak flow velocity has been depicted in Table 1. The results have been demonstrated for 70 bar, 80 bar, 90 bar and 100 bar inlet pressure conditions. Simulation results are represented in Table 1, 2, 3 and 4. For each pressure leak flow simulation has been observed for 3 sets of crack opening areas. Similarly, for rectangular cracks all such pressure conditions were simulated through the CFD module using applicable boundary conditions. The results have been depicted in table 2. More simulations were carried out for even higher pressures up to 130 bar for various crack configuration and pipe lengths to check their effects.

Table 1: Mass flux and maximum leak flow velocity data predicted for circular cracks

Inlet pressure (Bar)	Crack area (COA) (x 10^{-6}) m^2	Mass flux through crack (Kg/m^2 s)	Maximum velocity (m/s)
70	0.79	3.05	99.45
	7.07	3.23	119.50
	19.63	3.30	124.50
80	0.79	3.27	106.4
	7.07	3.46	127.80
	19.63	3.53	133.20
90	0.79	3.47	112.90
	7.07	3.68	135.60
	19.63	3.75	141.40
100	0.79	3.65	118.90

	7.07	3.88	143.00
	19.63	3.96	149.00

Table 2: Mass flux and maximum leak flow velocity predicted for rectangular cracks

Inlet pressure (Bar)	**Crack area (COA) (x 10^{-6}) m^2**	**Mass flux through crack (Kg/m^2 s)**	**Maximum velocity (m/s)**
70	2.00	3.16	665.70
	5.00	3.21	893.80
	8.00	3.23	986.00
80	2.00	3.39	689.60
	5.00	3.44	952.20
	8.00	3.46	1043.00
90	2.00	3.60	716.70
	5.00	3.64	986.80
	8.00	3.67	1091.00
100	2.00	3.80	742.40
	5.00	3.86	1032.00
	8.00	3.87	1139.00

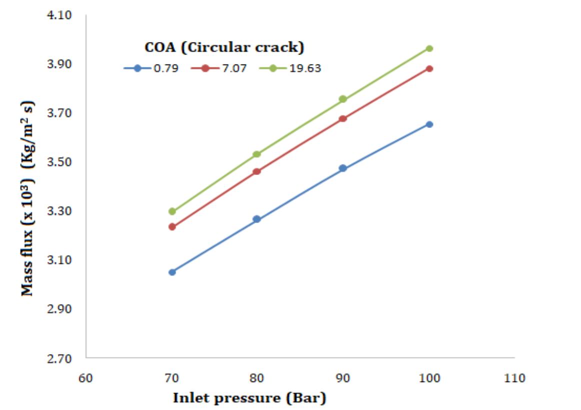


Figure 2. Mass flux variance with inlet pressure

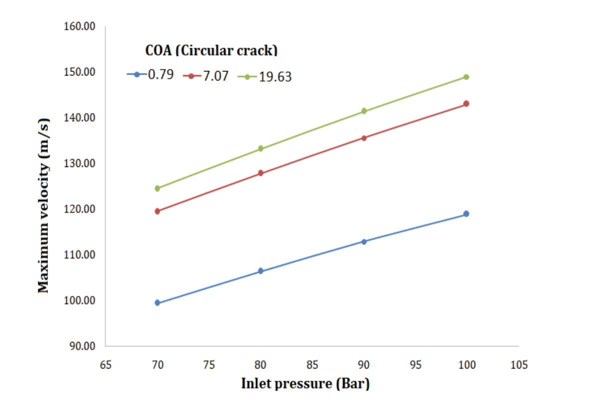


Figure 3. Maximum velocity variance with inlet pressure

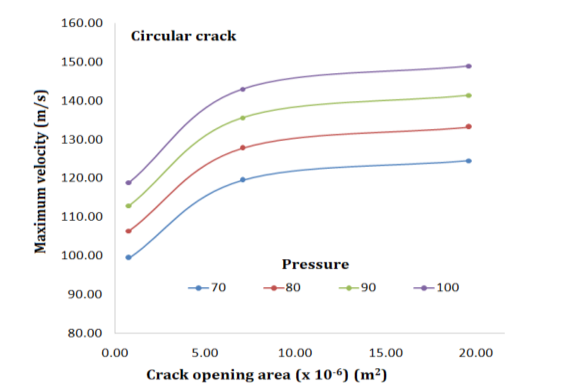


Figure 4. Maximum velocity variance with COA

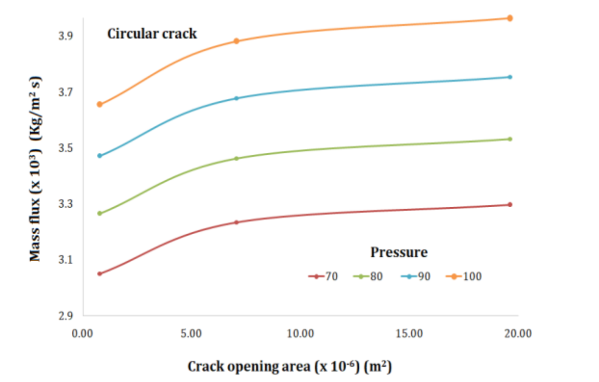


Figure 5. Mass flux variance with COA

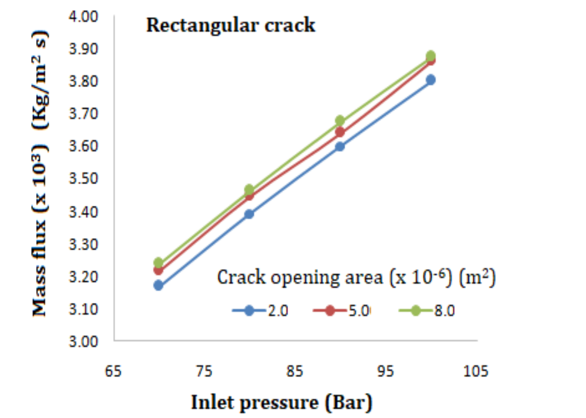


Figure 6. Mass flux variance with inlet pressure

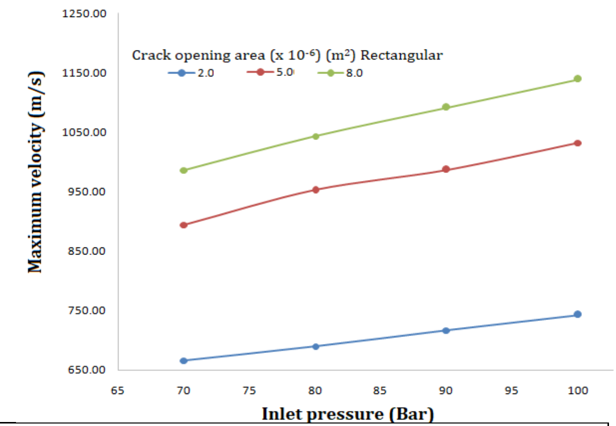


Figure 7. Variation of maximum velocity with inlet pressure

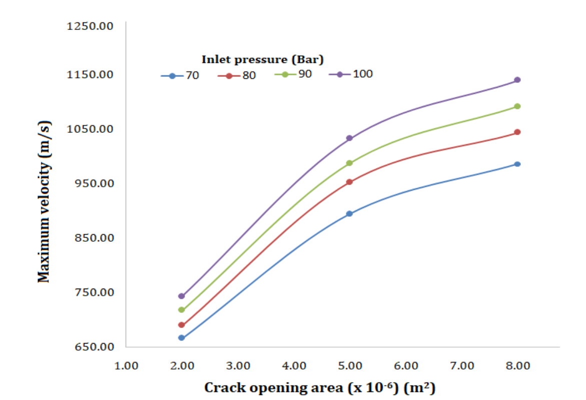


Figure 8. Maximum velocity variance with COA

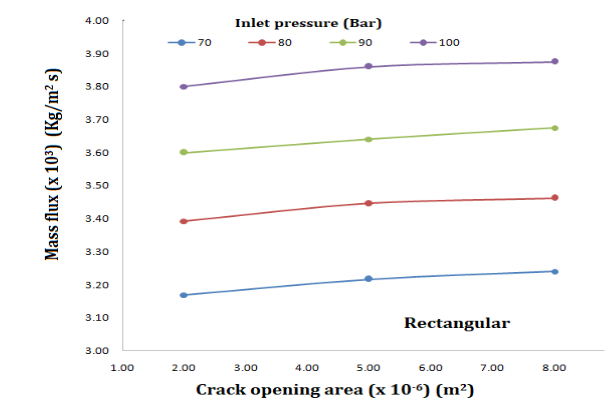
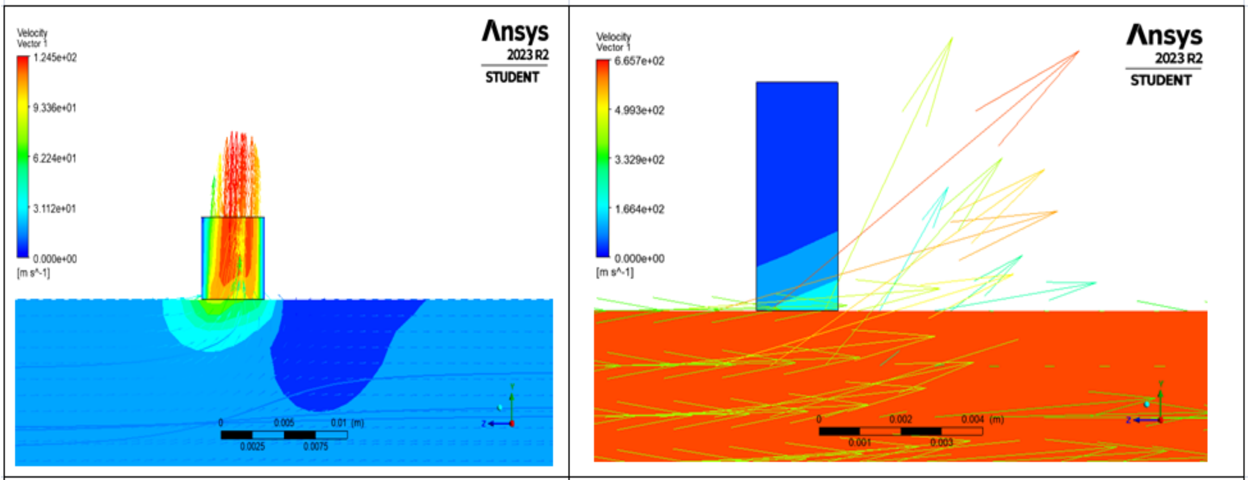


Figure 9. Variation of mass flux with COA

The velocity profiles presented below illustrate the velocity distribution along the cracked segment of the pipeline and within the outlet section. Across all simulated scenarios, this velocity profile remains consistent, depicting lower velocities along the crack wall and reaching maximum values at the center, as specified by the boundary conditions. The crack opening area has significant impact on mass flux through the crack. For a particular inlet pressure conditions this work put forward that with the increase in crack opening the mass flux also increases (fig. 5 & 9). The result of velocity on the crack opening area has been also been shown in the (fig. 4 & 8). With increase in crack opening areas an increasing trend is observed. There is an increase in mass flux and velocity with an increase in inlet pressure for a constant COA. There is an increase in mass flux and velocity with an increase in COA for a constant pressure as reported in several experimental works [9, 10]. Despite a constant pressure gradient between the inlet and main pipe outlet, the mass flux and velocity change with a change in inlet pressure. The plots show that the trends are similar for both the shape of the cracks as observed in the Figures 2, 3, 4 and 5 for circular cracks and in figures 6, 7, 8 and 9 for rectangular cracks. Results are analogous to the experimental findings with inlet pressure greater than 100 bar for changed pipe length under consideration. The predicted pressure and velocity trends conform in significant number of cases with major experimental findings reported in the literature [10-12]. Further works can be conducted including the effect of temperature at high pressure conditions as elaborated in several reported works [13-15] for better approximation of the industry conditions.



Circular crack	Rectangular crack

Figure 10: Velocity contour at the leakage zone for circular and rectangular crack geometry

IV. CONCLUSION

The analysis of results from CFD simulations conducted under varying conditions of inlet pressure, crack opening area, crack shape, and velocities at the crack outlet yields various insights. Under a consistent pressure gradient between the pipe inlet and crack outlet, both mass flux and velocity exhibit changes corresponding to alterations in inlet pressure. Moreover, the mass flux demonstrates fluctuations in response to variations in both inlet pressure and crack opening area showed proportional effects. These trends remain consistent across different crack shapes, as depicted in the leak flow behavior demonstrated in the work. In case of simulations done with higher pipe length and higher stagnation pressures, it was observed that the trends are independent of the length of pipe section under interest. These findings align closely with prior research data in this field. Furthermore, the study underscores the efficacy of CFD analysis methods in validating theoretical data within real-world scenarios.

REFERENCES

[1] Shammas N K and Al-Dhowalia K H 1993 Effect of Pressure on Leakage Rate in Water Distribution Networks, *Journal of King Saud University - Engineering Sciences* **5**(2) 1993, pp. 213-226.

[2] Shao Y, Yao T, Gong J, Liu J, Zhang T and Yu T 2019 Impact of Main Pipe Flow Velocity on Leakage and Intrusion Flow: An Experimental Study. *Water* **11**, 118. https://doi.org/10.3390/w11010118

[3] Sousa de CA and Romero OJ 2021 Influence of oil leakage in the pressure and flow rate behaviors in pipeline *Latin American Journal of Energy Research* **4(**1) pp. 17–29.

[4] Ali S, Hawwa MA and Baroudi U 2022 Effect of Leak Geometry on Water Characteristics Inside Pipes. *Sustainability* **14**, 5224. https://doi.org/10.3390/su14095224

[5] Manna S, Kundu A, Dey K, Roy S, Bhuit R and Ghosh S 2023 CFD simulation of circumferential crack in low pressure water pipelines, *Materials Today: Proceedings* **80** Part 2, pp. 806-812.

[6] Ghosh S and Saha S K 2021 Modeling and Simulation of Subcooled Coolant Loss Through Circumferential Pipe Leakage *ASME J of Nuclear Rad Sci*. **7**(3): 034503 (8 pages) https://doi.org/10.1115/1.4048477

[7] Mansour R B, Habib M A, Khalifa A, Youcef-Toumi K and Chatzigeorgiou D 2012 Computational fluid dynamic simulation of small leaks in water pipelines for direct leak pressure transduction, Computers & Fluids **57**, pp. 110-123.

[8] Dubyk Y 2019 Application of Probabilistic Leak-Before-Break for WWER-1000 Unit, *Procedia Structural Integrity* **22**, pp. 275-282.

[9] Henry R E and Fauske H K 1971 The Two-Phase Critical Flow of One-Component Mixtures in Nozzles, Orifices, and Short Tubes, *J. Heat Transfer*. **93**(2) pp. 179-187. https://doi.org/10.1115/1.3449782

[10] Amos CN and Schrock VE (1984) Two-Phase Critical Flow in Slits *Nucl. Sci. Eng.* **88**, pp.261-274

[11] Chang K S 1999 Uncertainty Analysis of Leak Rate Calculation through Pipes and Slits *Annals of Nuclear Energy* **26**, Issue 5, pp. 411–422

[12] Giot M 1994 Two-phase releases *Journal of Loss Prevention in the Process Industries* **7(**2) pp. 77–93.

[13] Henry RE 1970 The two phase critical discharge of initially saturated or subcooled liquid. *Nucl. Sci. Eng.* **41** pp. 336 - 342.

[14] John H, Reimann J, Westphal F and Friedel L 1988 Critical two-phase flow through rough slits, *International Journal of Multiphase Flow* **14** (2) pp. 155–174

[15] Rudland D. L, Wilkowski, G. and Scott P 2002 Effects of crack morphology parameters on leak-rate calculations in LBB evaluations. *International Journal of Pressure Vessels and Piping* **79**(2) pp. 99-102.